International Journal of Mechanical and Production
Engineering Research and Development (IJMPERD)
ISSN (P): 2249–6890; ISSN (E): 2249–8001
Vol. 9, Issue 6, Dec 2019, 263–278

CFD ANALYSIS OF A HEAT EXCHANGER USING ENHANCED WALL TREATMENT FUNCTION TO CAPTURE THE LAMINAR SUB-LAYER CLOSE TO THE WALL

K. P. V. KRISHNA VARMA[1], B. SRINIVASA VARMA[2] & K. RATNA KUMARI[3]

[1]*Assistant Professor, Department of Mechanical Engineering, CMR College of Engineering & Technology, Telangana, India*

[2]*Professor, Department of Mechanical Engineering, CMR College of Engineering & Technology, Telangana, India*

[3]*Assistant Professor, Department of Mechanical Engineering, CMR Technical Campus, Telangana, India*

ABSTRACT

Present work reports on analysis of effect of wall functions in a STHE on surface heat transfer. The inlet temperature of hot fluid, which is flowing in a pipe, is maintained at 333Kand inlet of cold fluid, flowing in shell is maintained at of 300K. The tube wall is conductive medium and is made of stainless steel, where shell tube is considered as a pure insulator. Water is considered as working fluid for this study. HTC of wall is considered as the variable of interest to analyze enhanced wall function and K-e standard wall function. Using enhanced wall treatment function, the velocity distribution and the temperature distribution near the wall regions for laminar sub layer are captured clearly, which is otherwise not possible with the standard wall function.

KEYWORDS: y+ Values, Navier Stokes Equation, Sub Layer & Laminar Flow

Received: Aug 29, 2019; **Accepted**: Sep 20, 2019; **Published**: Oct 24, 2019; **Paper Id**.: IJMPERDDEC201923

Original Article

1. INTRODUCTION

Turbulence is always dominated by higher shear stresses. As the flow proceeds, velocity gradient reduces and will be greater at the walls since shear stress is large. So, it can be concluded that at the walls, turbulence will be zero and flow remain laminar. This region near the wall is called as laminar sub-layer. There will be turbulence if we move from this region as the shear stress will tend to decrease. This particular layer is called as the buffer layer. Understanding nature of turbulence in this layer is very complex and capturing of turbulence or laminar sub-layer in this area is not possible with the available standard wall functions in Ansys. This can be possible with the usage of enhanced wall treatment function [1] [2]. Few researchers have done their work in this area. Some of the work done is quoted below. Fiuza1et.al. [3] carried out simulations for air flowing in a T-Junction channel. RANS equation is used along with k-€ model for a 2- dimensional flow. Reynolds Number, Moment ratios are considered as input parameters and the results obtained are compared using large eddy simulations. Jones et al.[5] proposed a new model for determining local turbulence viscosity for less Reynolds number. This model determined Wall boundary-layer also. Georgi et .al [6] provided an approach, which is used to any of the RANS Turbulence models. Analysis is done to find the behavior of RANS turbulence models. Mohd Ariff et.al [8] found an approach for turbulent flows using y^{+}values while selecting the grid configuration. Accordingly, turbulent models are investigated. Spart-Allmarahs model gave good results which are very close to values in the literature. This Defraeye et.al [9] proposed a process to determine user oriented temperature wall function. There is a reduction of deviations from the standard wall functions, which is shown by this process. Gharbiet.al [10] compared different models of treatment near the

wall. The attained values are differentiated with that of DNS.M. Holling [11] Analysis is carried out to study turbulent natural convection for heated and cooled walls. Modifications are done in momentum equation. Salim et.al [12] developed a procedure to deal with the turbulence across a 2-dimensional surface mounted with an obstacle. Y^+values are used in selecting configuration of the grid and corresponding models for turbulence. The simulations are carried out for both distributed and undistributed flows. For both the cases, a range of y+ values are determined. Juan Gonzalo Ardila Marín et.al [14] tried to validate the numerical models of heat exchangers with the data available in the literature. This study dealt with flow inside the curved tube with corrugations, external flow in twisted curved tubes. The results depicted an increment in zeat flux with the change in geometry of heat exchangers. Mustafa Kemal Isman et.al [15] explored the flow under turbulence condition with backward-facing method (BFS). Uniform as well as non- uniform profiles of velocity are considered as boundary conditions.

2. NUMERICAL MODEL

Figure 1 shows the two-dimensional and three-dimensional models of the heat exchangers considered for the simulation [4]. The length of the heat exchanger is 1.4m; the diameter of the inner tube is 0.017m with wall thickness of 0.02m. The diameter of shell is 0.06m. **Table 1** shows the operating conditions under which, heat exchanger is simulated.

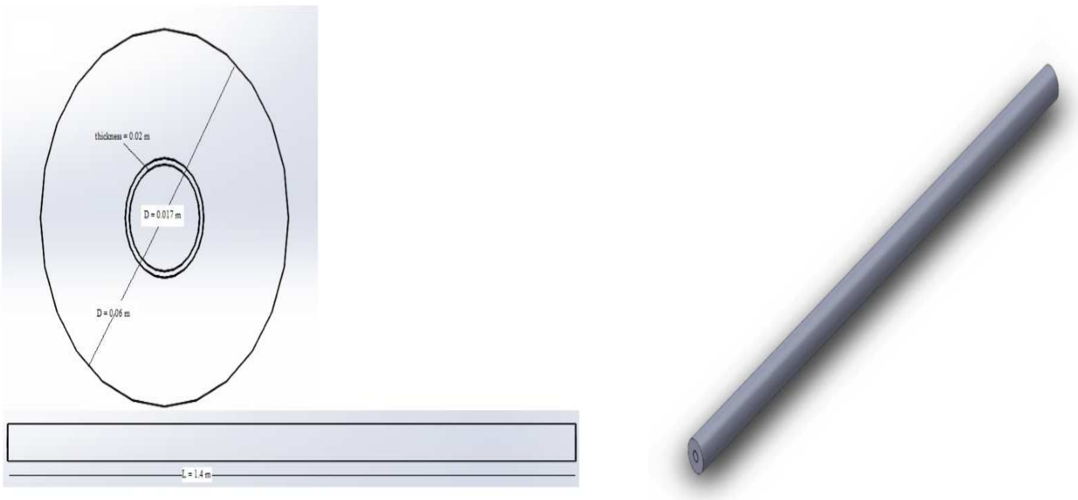

Figure 1: Configuration of the Shell and Tube Heat Exchanger.

Table 1: Operating Conditions

Water Density	1000 kg/m3
Cold water temperature	303 K
Hot water temperature	333K
Cold water mass flow rate	6 LPM
Hot water mass flow rate	6 LPM

2. 1. Mathematical Model

- Heat transferred to the cold water

$$QC=m_c c_{pc} (T_{co} - T_{ci}) \text{ j/s} \tag{1}$$

- The heat transfer rate from the hot water in the annulus side

$$Qh=m_h c_{ph} (T_{hi}-T_{ho}) \text{ j/s} \tag{2}$$

- Overall rater of heat transfer can be calculated from

Qavg =AiUwΔTLMTD (3)

Where,

$$\Delta T_{LMTD} = \frac{(T_{hi}-T_{co})-(T_{ho}-T_{ci})}{\ln(\frac{T_{hi}-T_{co}}{T_{ho}-T_{ci}})} \quad (4)$$

m_c = Mass flow rate, cold fluid

m_h = mass flow rate, hot fluid

Tc,o = outlet temperature, cold fluid

Tc,i = inlet temperature, cold fluid

Th,o = outlet temperature, hot fluid

Thi = Temperature of hot inlet

U' = overall htc for the wall

Cp,h = specific heat, hot fluid

Cp,c = specific heat, cold fluid

These equations govern the heat transfer between two fluids with the wall in between them.

3. GOVERNING EQUATIONS

The governing equation defines the type of flow and the field variables that have to be calculated. Typical governing equations pertaining to incompressible flow are continuity, momentum equations. Equations pertaining to energy as well as mass and momentum are given by Navier-stokes.

3.1. Conservation of Mass

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \quad (5)$$

Equation (5) shows equation for conservation of mass for incompressible and compressible flows. For any source defined by the user and for dispersed second phase, Sm is the added mass.

3.2. Conservation of Momentum

Conservation of momentum in a stationary frame is given by:

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \quad (6)$$

Where, p denotes pressure under static conditions, $\overline{\overline{\tau}}$ denotes stress tensor and pg & F are the gravitational body force and external body forces, respectively. F comprises of sources that are depending on other models such as porous-media and sources defined by user.

3.3. Conservation of Energy

In a mathematical model, to solve for temperature distribution, Navier Stokes expression can be used and is given by

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = \nabla \cdot \left(k_{\text{eff}} \nabla T - \sum_j h_j \vec{J}_j + (\overline{\overline{\tau}}_{\text{eff}} \cdot \vec{v}) \right) + S_h \tag{7}$$

In above equation, K_{eff} denotes conductivity (effective) and Jj represents flux (diffusion) of species j. The terms existing to the right side denote transfer in energy due to conduction as well as diffusion due to species and dissipation due to viscous media.

In equation8, E is given as follows

$$E = h - \frac{p}{\rho} + \frac{v^2}{2} \tag{8}$$

Where, enthalpy(sensible) h for incompressible flow and is given by

$$h = \sum_j Y_j h_j + \frac{p}{\rho} \tag{9}$$

In the above equation Yj is

$$h_j = \int_{T_{\text{ref}}}^{T} c_{p,j}\, dT \tag{10}$$

The value used for Trefin the sensible enthalpy calculation depends on the solver and model in use. For the pressure-based solver, it is 298.15k

Energy equation in solid regions is given below:

Since in this problem the inner tube pipe is solid, Ansys fluent solves this region by using the below equations

$$\frac{\partial}{\partial t}(\rho h) + \nabla \cdot (\vec{v} \rho h) = \nabla \cdot (k \nabla T) + S_h \tag{11}$$

where ρ = density

h = sensible enthalpy, $\int_{T_{\text{ref}}}^{T} c_p dT$

k = conductivity

T = temperature

S_h = volumetric heat source

4. NUMERICAL PROCEDURE

Numerical Analysis comprises of the following steps as shown in **figure 2.** The first step called as preprocessing, which involves creation of fluid domain from physical problem, Meshing using appropriate techniques, checking for quality using quality criteria, Grid independent test and exporting the mesh to solver. The second most important step in numerical procedure is solving, this step involves selection of proper solver like pressure based or density based on Mach number and selecting solver type weather SIMPLE, SIMPLEC, COPULED etc., and defining Spatial Discretization orders etc. The final step in Numerical procedure is post-processing. In this step, the solution obtained for filed variables are analyzed with the help of contours, plots, vectors and stream lines etc.

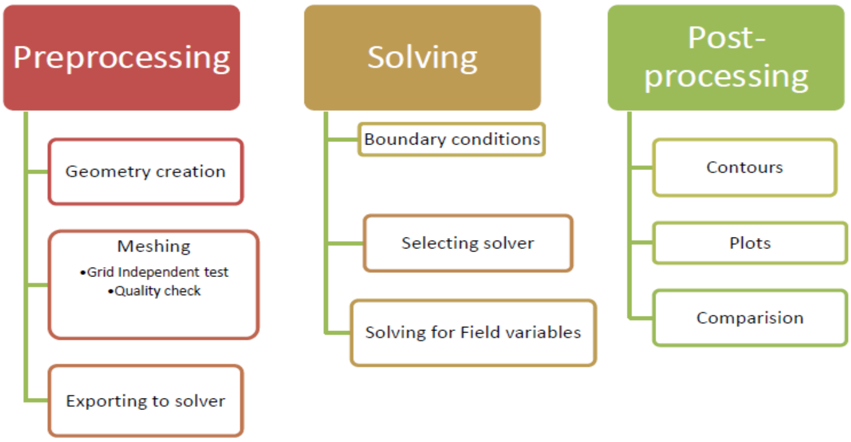

Figure 2: Numerical Procedure.

A 3-D Numerical model with required dimensions is developed using ICEM CFD. This geometry is meshed in two different topologies Tetra (unstructured mesh) using global and part mesh parameters and HEXA (Structured mesh) using blocking and O-grid with edge parameters [5]. ICEM CFD is one of the most powerful tools, which uses intelligent blocking technique to generate Hexa grids [2]. Hexa grids are more efficient in solution calculations. The solution time with Hexa grids will consume less time compare with the Tetra grids for the same results. However, generation of Hexa grids in complex geometries is challenging. It requires proper understanding of blocking technique, and one should know how to slice the complex geometry in blocks to generate structured quads on surface and hexahedral grids in the volume. The **figure 3** shows the O-grid generation in a pipe, after blocking and associating edges to the geometry, O-grid blocks are generated to capture small topologies inside the larger geometry. In this problem, larger geometry is shell, and the smaller geometry is tube. To capture Hexa inside the tube O-grids are generated. **Figure 4** shows the mesh generated after O-grid meshing. Quality of the mesh attained in this case is ranging from 0.7 to 1.0. Mesh having quality more than 0.3 and approaching 1.0 is considered as a mesh of good quality**. Figure 5** shows the TETRA mesh. The elements obtained for Tetra mesh for same quality criteria as Hexa mesh is 2917754, which is 6 times more than the Hexa mesh. This concludes that the solution time with Tetra mesh wills increases 6 times more than the solution with Hexa meshes. Therefore, in this simulation, Hexa mesh with 0.7 quality criteria is selected for the solution. The **figure 6** shows the quality criteria chosen for the solution, which ranges from 0.7 to 1.

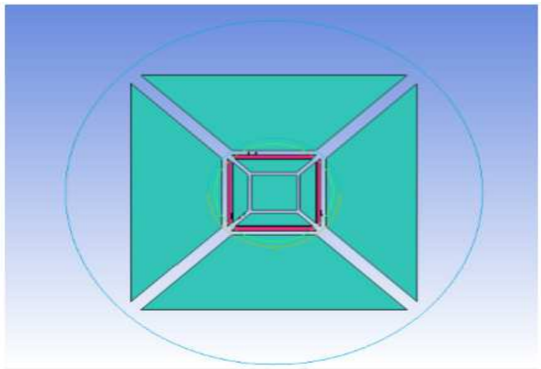

Figure 3: O-grid Generation.

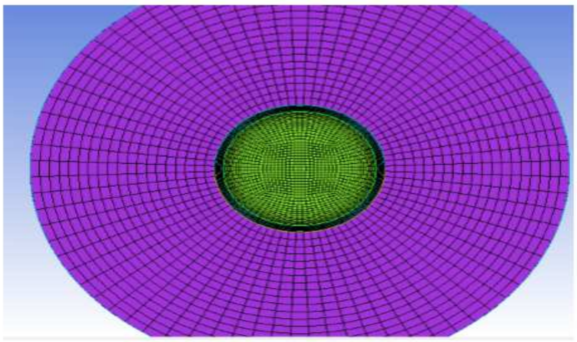

Figure 4: Fine HEXA Mesh.

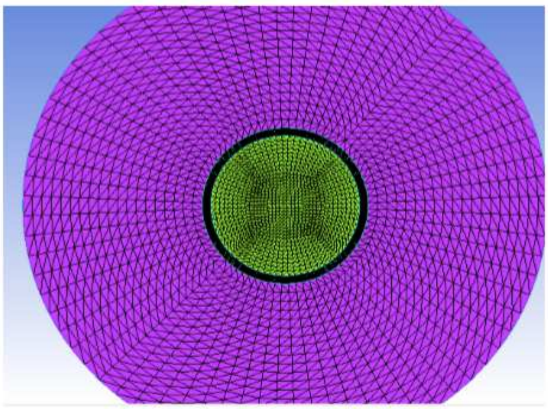

Figure 5: Tetra Mesh.

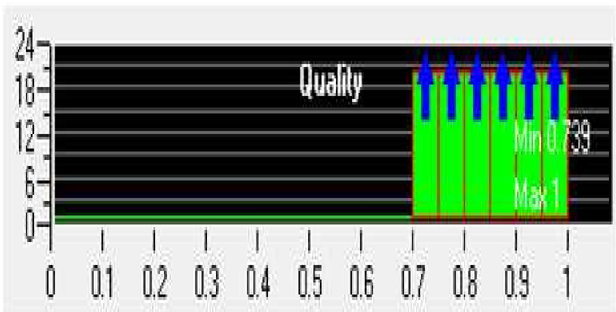

Figure 6: Mesh Quality Criteria.

5. GRID INDEPENDENT TESTS

Grid independent test is the necessary step for any Numerical problem to get correct results. The results always should be independent of mesh i.e. results should not vary as Mesh size varies. In this problem, a grid independent test is performed for laminar flow with Re 100 and Turbulent flow with Re 105. This test is performed based on axial velocity and entrance length outlets by varying grid sizes. **Table 2** shows the Variation of the sizes and their results.

Table 2: Grid Independent

Mesh Type (Re 100)	Size	Quality	Heat Transfer Coefficient
Coarse	157592	0.4	408.32
Medium	256227	0.7	420.55
Fine	472777	0.9	425.81

% Error in Heat transfer coefficient

% error = (420.55−410.32)/420.55 *100 = 2.90% < 5%

% error = (434.81−420.55) /434.81∗ 100 = 1.23% < 5%

From the above grid independent test, it has been observed` that for the fine mesh the quality was 0.9 and the percentage error is below 5%, hence this mesh was selected for solution.

5.1. Solving

Solving is the second main step in numerical procedure, this includes selection of solver, defining boundary conditions and solving for field variables.

Ansys Fluent 19.2 is used to solve the problem numerically. A pressure-based solver is chosen to solve for continuity, momentum and energy equations sequentially. This solver Solves the momentum equations, one after another, using the recently updated values of pressure and face mass fluxes. Pressure based solver is memory efficient, since the discretized equations need only be stored in the memory one at a time. In this, water is working fluid and the tube material is solid steel. The properties of these materials are shown in the table 3.

Working Fluid and solid Properties are given in **table 3**.

Table 3: Fluid Properties

Material	Density (kg/m3)	Thermal Conductivity (w/m-k)	Cp (Specific Heat) (j/kg-k)
Water	998.2	0.6	4182
Steel	8030	16.27	502.8

5.2. Models

Since the problem deals with heat transfer, energy model is used. Reynolds number is significantly high for this problem, which involves using of standard two equation models i.e., K-ε model also called as kinetic dissipation model. Along with this model, a standard wall function and enhanced wall functions are chosen for two different cases, one with standard wall function and other with enhanced wall function. Standard wall functions will capture only the log region of boundary layer where y+ value lies from 30 to 300.This model fails to capture the laminar sub-layer which is very close to the wall region. The enhanced wall function will capture the laminar sub-layer in the boundary layers, therefore this wall functions is used. Enhanced wall function will capture the y+ values from 3 to 10. **Table 4** shows the types of models used

Table 4: Models

Type	Models
Heat transfer	Energy
Viscous	Standard K-ε model
Wall functions	1. Standard wall function 2. Enhanced wall functions

5.3. Cell Zone Conditions

The computational domain is divided into two cell zones, namely shell-fluid which defines the liquid zone and this zone is assigned with liquid (water) and Tube-solid which defines the solid zone; in this case, solid zone is assigned with steel.

5.4. Mesh Interface

There are two Interfaces in this problem; the first interface is between the hot fluid inside tube and inner wall of the tube. The second interface is between the outer wall of the tube and the cold fluid inside the shell. These interfaces are called as coupled walls, where heat transfer between them takes place.

5.5. Boundary Conditions

In this problem, the typical boundary conditions include inlet and outlet conditions for both Cold and Hot liquids and wall conditions. The **Table.5** shows the boundary conditions used in the problem.

Table 5: Boundary Conditions

Type	Momentum	Thermal
Cold Inlet	Velocity 0.0353 m/s	Temperature 300 k
Cold Outlet	Outflow	
Out flow Hot inlet	Velocity 0.0353 m/s	Temperature 333 k
Hot Outlet	Outflow	
Shell wall	No slip condition	Adiabatic wall
Tube wall	No slip condition	Conductive wall

5.6. Solution

SIMPLE scheme is selected for Pressure-velocity coupling, see **table. 6,** as this scheme solves the mass and momentum equations in the first place and after converging of mass and momentum, it solves for energy equations. In this way, the wall heat transfer coefficient will be accurate as it will solve for energy equation, separately

Table 6: Scheme Type

Coupling	Scheme
Pressure -Velocity	SIMPLE

In spatial discretization mentioned in table 7, green-gauss node-based gradient is used, as this method is suitable for getting accurate results. The second order spatial discretization is used for Pressure-Momentum and Energy, whereas first order is used for the turbulent kinetic energy and turbulent dissipation.

Table 7: Spatial Discretization

Type	Spatial Discretization
Pressure	Second Order
Momentum	Second Order
Energy	Second Order
Turbulent Kinetic energy	First Order
Turbulent Dissipation	First Order

5.7. Solution Controls

Solution controls are the parameters used to maintain the solution from diverging. Pressure, density, turbulent kinetic energy, momentum and body force the parameters responsible for divergence. The **Table 8** shows the under relaxation factor values given for this parameter.

Table 8: Under Relaxation Factors

Parameters	Under-Relaxation Factors
Pressure	0.3
Density	1
Momentum	1
Body force	0.7
Turbulent kinetic energy	0.8

5.8. Solution Initialization

Hybrid initialization is used for the problem, as this type of initialization will guess the results of the problem based on the input boundary conditions.

5.9. Post-Processing

Post processing includes Analysis of results, discussion of results and comparison with the use of contours, plots and stream lines that are obtained from the solution. In this problem, the parameters are temperature, velocity, wall surface heat transfer coefficient. The contours of temperature, velocity and wall surface heat transfer coefficient are studied.

6. ANALYSIS OF RESULTS

6.1. Standard Wall Function

The outlet temperature obtained for the cold fluid is 303.42K, and the outlet temperature of the hot fluid is 329.50K. The overall heat transfer coefficient of the solid tube from inner wall to outer wall is 239.71 W/m2-K. The Y+ value near the wall obtained is 42

6.2. Enhanced Wall Function

The outlet temperature obtained for the cold fluid is 304.39K, and the outlet temperature of the hot fluid is 328.81K. The overall heat transfer coefficient of the solid tube from inner wall to outer wall is 425.81 w/m2-k. The Y+ value near the wall obtained is 3.

6.3. Results and Discussions

The **figure 8** and **9** shows the temperature distribution in the heat exchanger with standard wall function in cut section view and cross section view, near the hot fluid inlet region. From the figure.8, it is seen that due to high temperature of the fluid, it is evident that there is constant heat conduction near the wall of solid tube which is not reliable in real scenario. The **figure 10** and **11** shows the same phenomenon near the outlet region, it is evident that due to heat transfer from hot fluid to cold fluid the temperature of the hot fluid reduced, this can be seen via variation in the color. The velocity variation near inlet region and outlet region can be seen in the **figure 12** and **13**. Near the walls, the velocity is zero due to no slip condition. Near the inlet, the velocity is low and as it progressed the velocity remained constant after it was fully developed indicating high velocity in the middle of the tube, which can be seen in the **figure 14.** The **figure 15** and **16** shows the conduction in the wall, which can be Cleary seen from the **figure 17,** as there is a sudden change in temperature, which

concludes that standard wall function failed to capture accurately near the walls. Enhanced wall function temperature distribution near inlet in cut section and cross section is seen in the **figure 18** and **19**. It is observed that there is constant distribution of near the wall region; this indicates that enhanced wall function captured the field variables in the laminar sub-layer of boundary layer. The same phenomenon of constant temperature is observed near the outlet region of hot fluid. This can be seen in the **figure 20** and **21**. Even the velocity distribution near the wall is well captured by the enhanced wall treatment. From **figure 22** and **23**, one can see the constant variation in the velocity. There is no sudden change in the velocity was observed.

From wall conduction, it is patently observed that there is constant decay in wall temperature unlike in standard wall function, as sudden change in temperature is observed. The **figure 24** and **25** shows the constant decay in temperature phenomenon. The **figure 26** and **25** shows the plots of comparison of wall surface temperature and wall heat transfer coefficient of standard wall function and enhanced wall function. From both these plots it is evident that higher surface temperature and wall heat transfer coefficient are obtained with enhanced wall function.

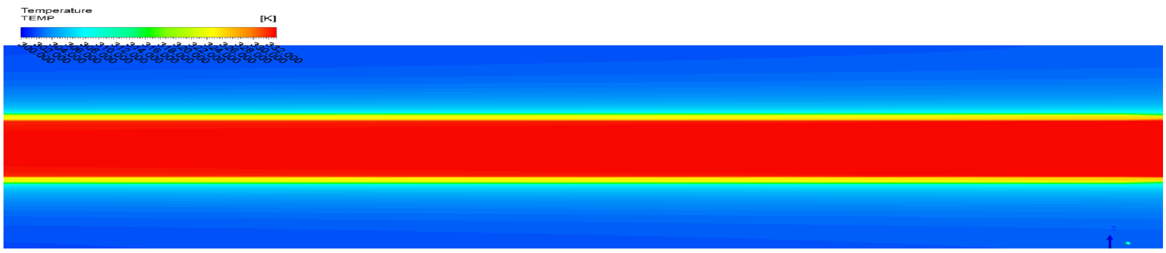

Figure 8: Temperature Near Inlet (Standard Wall Function).

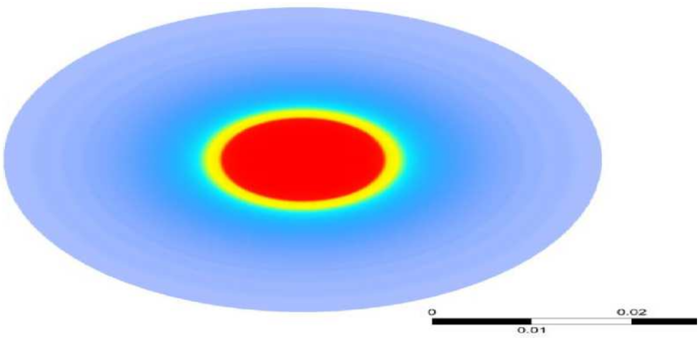

Figure 9: Cross Temperature Near Inlet (Standard Wall Function).

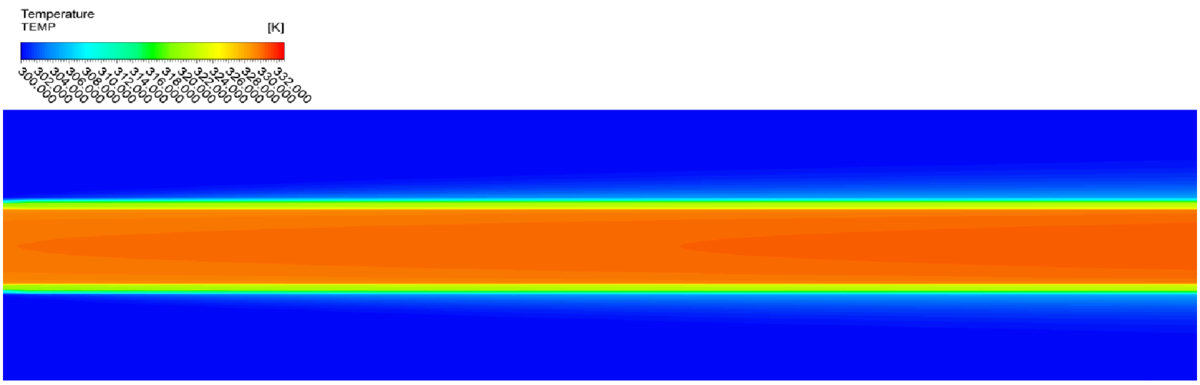

Figure 10: Temperature Near Outlet (Standard Wall Function).

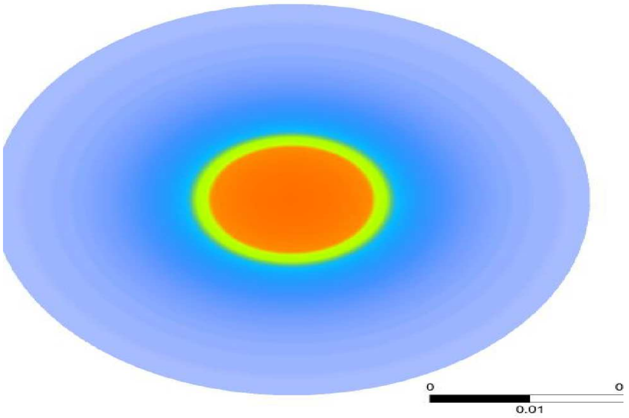

Figure 11: Cross Section Temperature Near Outlet (Standard Wall Function)

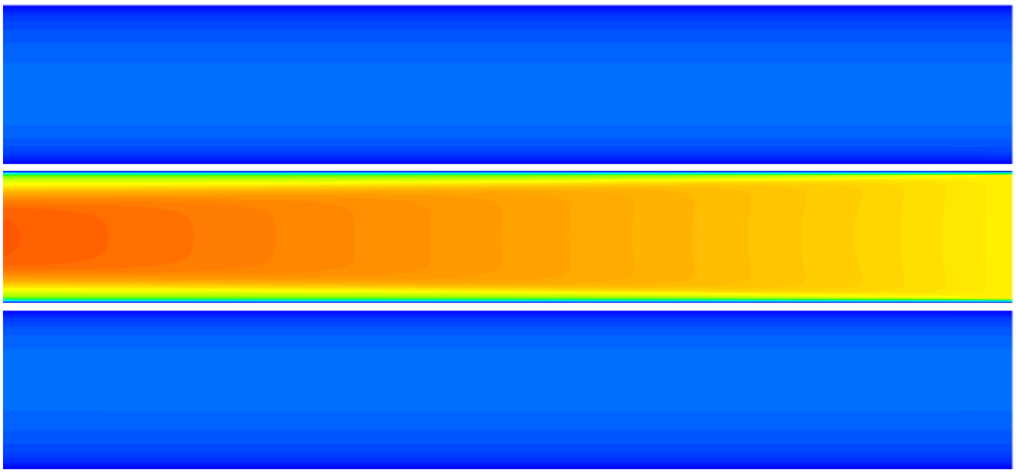

Figure 12: Velocity Near Inlet (Standard Wall Function).

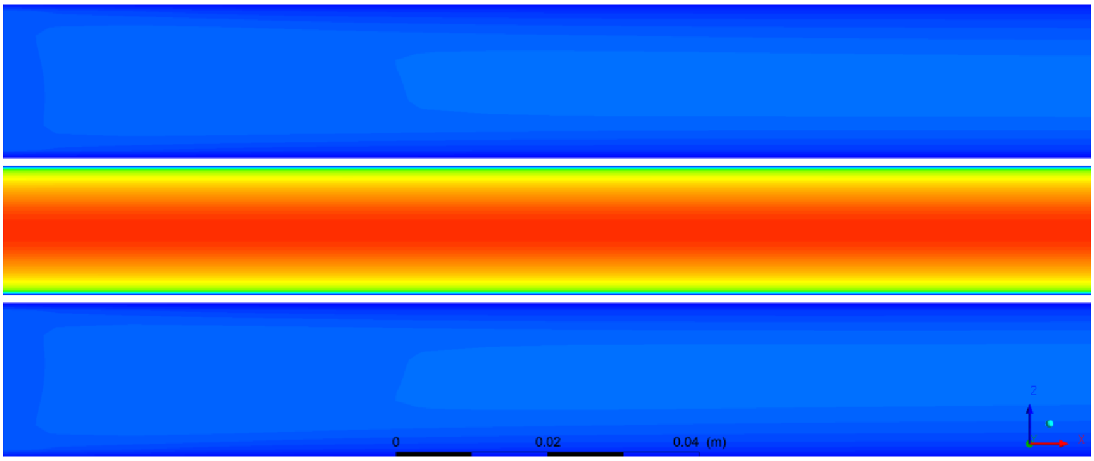

Figure 13: Velocity Near Outlet (Standard Wall Function).

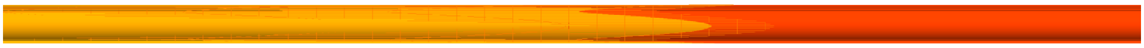

Figure 14: Conduction Through Wall Near Inlet Region (Standard Wall Function).

Figure 15: Conduction Through Wall Near Outlet Region (Standard Wall Function).

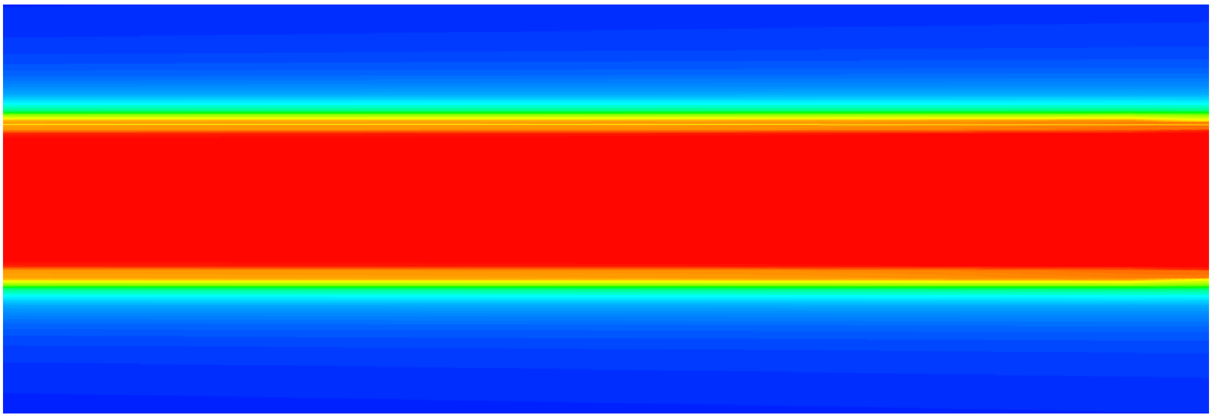

Figure 16: Temperature Near Inlet (Enhanced Wall Function).

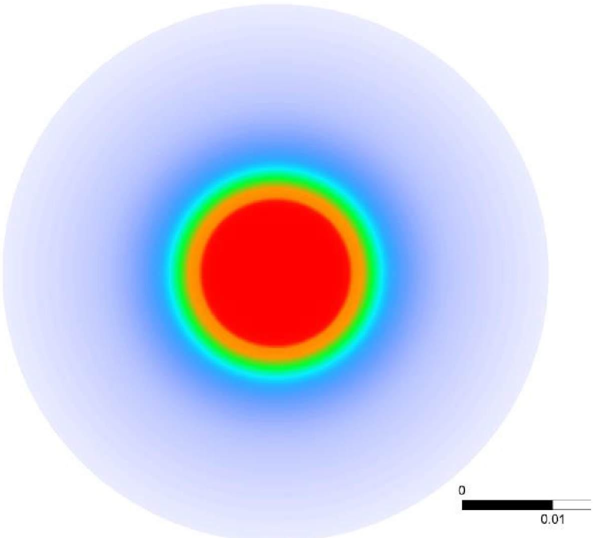

Figure 17: Cross Section Temperature Near Inlet (Enhanced Wall Function).

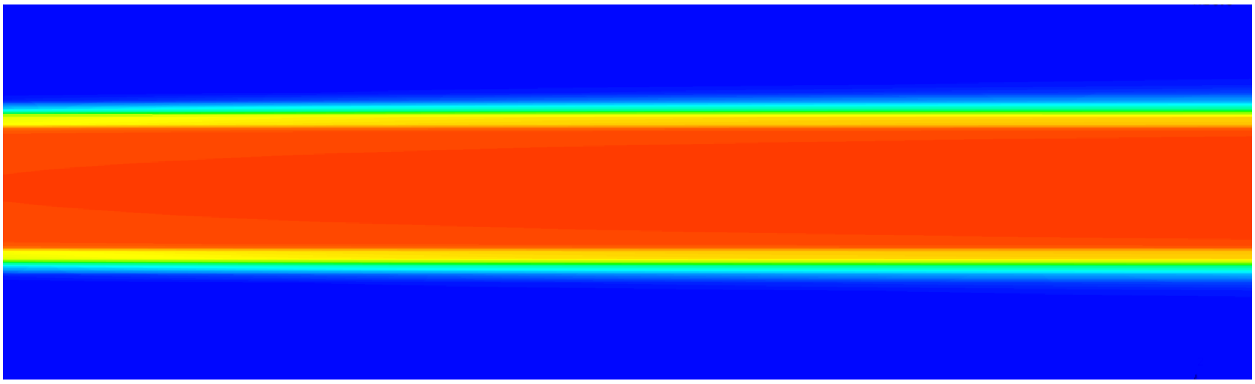

Figure 18: Temperature Near Outlet (Enhanced Wall Function).

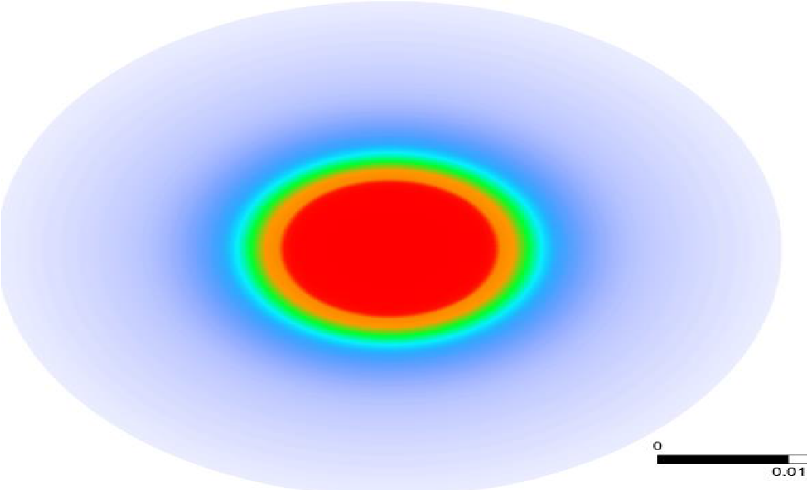

Figure 19: Cross Section Temperature Near Outlet Region (Enhanced Wall Function).

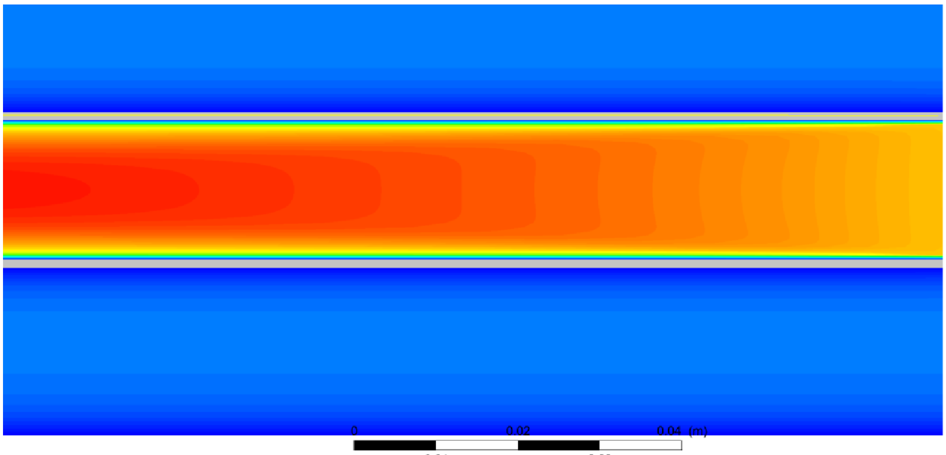

Figure 20: Velocity Near Inlet Region (Enhanced Wall Function).

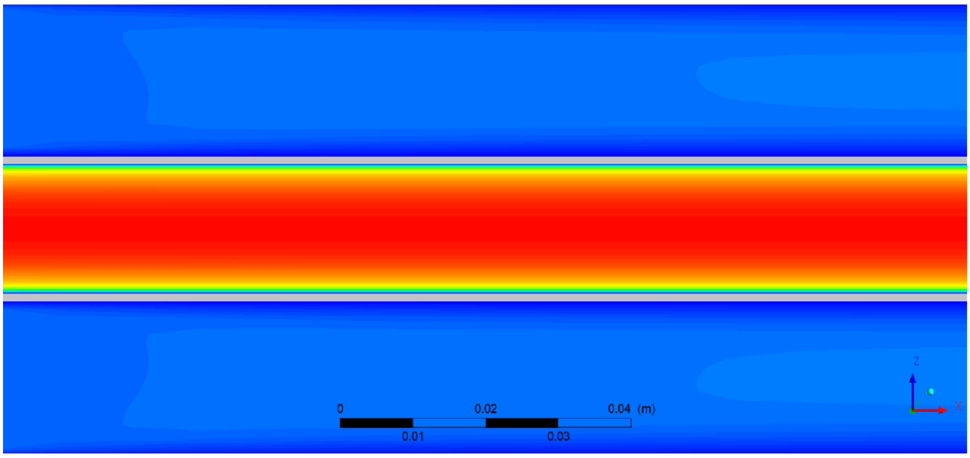

Figure 21: Velocity Near Outlet Region (Enhanced Wall Function).

Figure 22: Conduction Through Wall Near Inlet Region (Enhanced Wall).

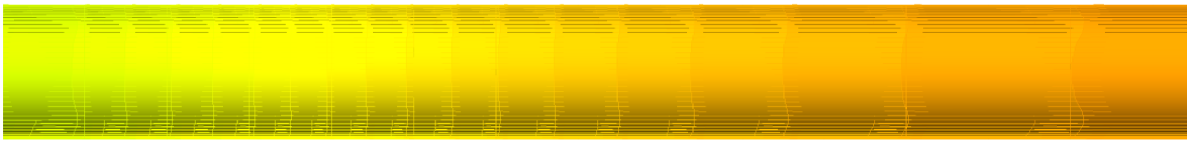

Figure 23: Conduction Through Wall Near Outlet Region (Enhanced Wall).

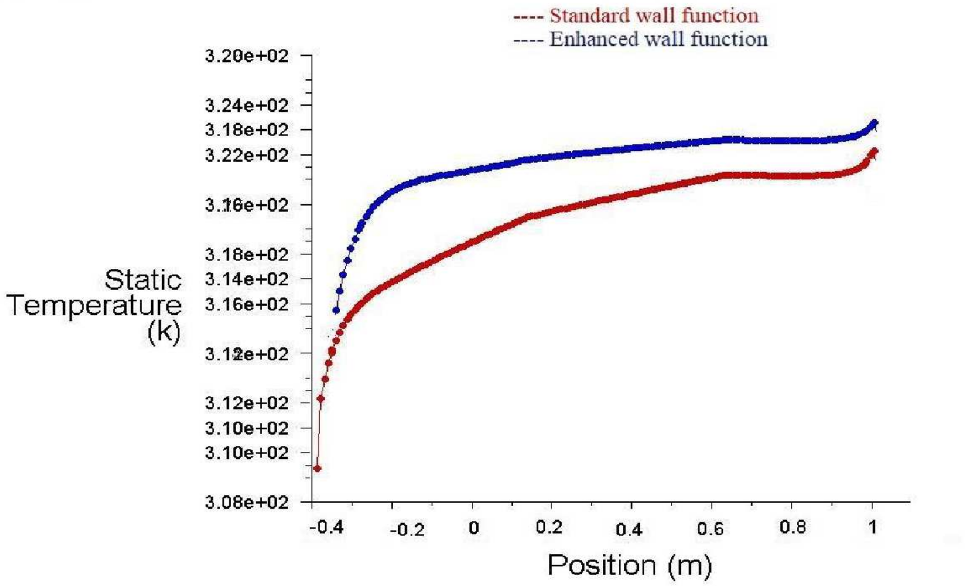

Figure 24: Surface Temperature Comparison.

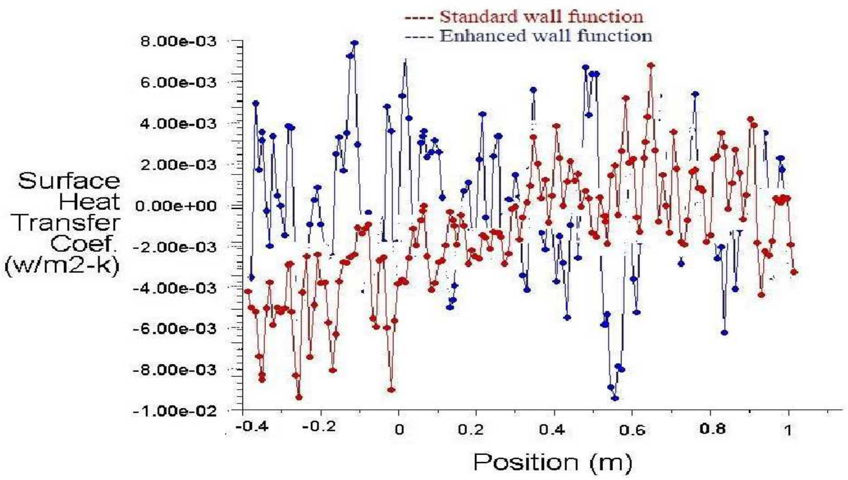

Figure 25: Surface Heat Transfer Coefficient Comparison.

7. CONCLUSIONS

In conclusion, Hexa mesh can solve the numerical procedure with less time as it can be seen from the meshing section, to generate same quality of TETRA mesh. It can be observed that in HEXA mesh, the elements count has increased nearly six times. This concludes that HEXA mesh is better than TETRA mesh for longer computational domains.

From the analysis and discussion of results section, we can conclude that enhanced wall functions capture laminar sub-layer in the boundary layer and solve the field variables, which are very near to the wall. There is no sudden decrease in wall temperature, observed in the enhanced wall function unlike standard wall function, which failed to capture viscous sub layer when a sudden change in temperature gradient is noticed. This concludes that Enhanced wall function should be used for the flow problems, where field variables near the wall are crucial such as heat transfer problem, in this case.

REFERENCES

1. *ANSYS FLUENT Theory guide, 2018, ANSYS, Inc*
2. *ANSYS ICEM CFD Tutorial Manual, 2018, ANSYS, Inc*
3. *Comparison of K-E Turbulence Model Wall Functions Applied on a T-Junction Channel Flow, G. C. C. Fiuza1, A. L. T. Rezende, International Journal of Engineering Research & Science (IJOER) - January- 2018*
4. *Holman, J.P., Heat Transfer, 5TH ed., McGraw Hill book Company*
5. *Jones, W. P.; Launder, B. E. Prediction of laminarization with a two-equation model of turbulence. International Journal of Heat and Mass Transfer. v. 5, p. 31–34,*
6. *GeorgiKalitzin, Gorazd Medic, GianlucaIaccarino, Paul Durbin, Near wall behavior of RANS models and implications for wall functions, Journal of Computational Physics, 2005, 265–291*
7. *Khan, S. A., Fatepurwala, M. A., & Pathan, K. N. (2018). CFD analysis of human powered submarine to minimize drag. Ratio (L/D), 4, 5.*
8. *Mohd ARIFF, Salim M. SALIM and Siew Cheong CHEAH, Seventh International Conference on CFD in the Minerals and Process Industries CSIRO, Melbourne, Australia 9–11 December 2009*
9. *Thijs Defraeye, Bert Blocken and Jan Carmeliet, An adjusted temperature wall function for turbulent forced convective heat transfer for bluff bodies in the atmospheric boundary layer,2011,46(11), 2130–2141*
10. *Najla El Gharbi, RafikAbsi, Ahmed Benzaoui, E.H. Amara, Effect of near-wall treatments on airflow simulations, International Conference on Computational Methods for Energy Engineering and Environment, ICCM3E, 20–22 November 2009, pp.185–189.*
11. *M. Holling, H. Herwig, Computation of turbulent natural convection at vertical walls using new wall functions*
12. *Salim.M. Salim, and S.C. Cheah, Wall y+ Strategy for Dealing with Wall-bounded Turbulent Flows, Proceedings of the International Multi Conference of Engineers and Computer Scientists 2009 Vol. II IMECS 2009, March 18–20, 2009, Hong Kong*
13. *QASHQAEI, A., & ASL, R. G. (2015). Numerical Modeling And Simulation Of Copper Oxide Nanofluids Used In Compact Heat Exchangers. International Journal of Mechanical Engineering, 4 (2), 1, 8.*
14. *Juan Gonzalo ArdilaMarín, Diego Andrés HincapiéZuluaga, Julio Alberto Casas Monroy, TECCIENCIA, Vol. 10 No. 19, 49–60, 2015.*

15. *Mustafa Kemal Isman, Investigation of inlet effects on backward-Facing step flow prediction, Transactions of the Canadian Society for Mechanical Engineering, Vol. 40, No. 3, 2016, pp.317–329.*

AUTHORS PROFILE

K.P.V.Krishna Varma: Assistant Professor, Department of Mechanical Engineering, CMR College of Engineering & Technology, Telangana, India. He published 8 papers in International Journals. He published 3 papers in Conferences and 4 papers in National Conferences

Dr. B.srinivasa Varma: Ph.D (Mechanical Engineering) from Regional Engineering College, Warangal, (presently NIT, Warangal) for the thesis titled "Development and Performance Evaluation of Alumina based Ceramic Cutting tools for machining Certain ferrous Materials" by Kakatiya university, Warangal in August 1999. Awarded Research fellowship by government of India (MHRD). He published 08 papers in International Journals and 10 papers in Conferences Presentations

K. RATNA KUMARI Assistant Professor, Department of Mechanical Engineering, CMR Technical Campus, Telangana, India. She published 8 research papers. Reserach work in Experimental studies on the effect of lignocellulosic biofuels obtained by catalytic hydrogenation of furfurals and value added chemicals on performance of compression ignition engine.